FREE-SURFACE PROTOTYPE PROBLEMS SUITABLE to INVESTIGATE PARTICLE METHODS

David LE TOUZÉ[a] & Andrea COLAGROSSI[b]

[a] École Centrale Nantes – Laboratoire de Mécanique des Fluides / Équipe EHGO
1 rue de la Noë, BP 92301, 44321 NANTES Cedex 3, France – david.letouze@ec-nantes.fr

[b] INSEAN, The Italian Ship Model Basin, Via di Vallerano 139, 00128 ROMA, Italy
a.colagrossi@insean.it

ABSTRACT

In this paper, an in-depth study of the SPH method is achieved on dedicated free-surface prototype problems. These rather critical prototype problems are believed to be suitable test cases to get through when building a SPH model. The goal is to investigate numerical aspects of this method that are little mentioned in literature. In particular, a specific attention is given to the dynamic part of the solution, *i.e.* the local hydrodynamic load prediction. The role of numerical errors in the development of acoustic frequencies in the pressure signals is discussed, as well as the influence of the choice of the sound velocity. The convergence of this method is heuristically proved on these non-linear prototype tests, showing at the same time the very satisfactory level of accuracy reached by our model. The advantages and drawbacks of using a higher-order form of the integral interpolation are also discussed. As well, it is shown that the same kind of tests are useful to investigate three dimensional versions of SPH solvers.

INTRODUCTION

At the last workshops, the SPH method has been shown to successfully resolve marine and coastal hydrodynamic problems, such as breaking wave patterns around vessels [1], two-phase flows [2], long-time sloshing evolutions [3], or water entry and floating-body/wave interaction problems [4]. However, the goal of the present paper is not to show the abilities of our SPH model in terms of applications, but rather to investigate more in depth various of its numerical aspects. In almost all the published SPH papers, authors give interest only to the kinematic part of the solution. Nonetheless, the evaluation of the local hydrodynamic loads is crucial when applying any numerical model to real applications in the field. As well, regarding the numerical mechanisms involved in the SPH method, people generally show either simple theoretical studies, or complicated results whose errors are difficult to analyse, and error factors practically impossible to determinate. Hence, there is a need for 'prototype' tests to validate actual numerical models. These tests must be chosen not too complicated, so as to have reference solutions to compare with and to avoid mixing too many physical aspects at the time, but chosen to be critical with respect to one or another aspect of the method. The proposed two- and three-dimensional prototype problems are believed to constitute suitable test cases to get through when building a SPH code (and more generally any free-surface solver). Through them, it is possible to discuss of aspects of the SPH method little highlighted in literature, especially: the role of the weak-compressibility and the numerical errors in the oscillations occurring in SPH pressure signals; the influence of the choice of the sound velocity; the importance and the effect of some numerical tools applied; the Lagrangian character of the method; and finally, the advantages and drawbacks of using a higher-order form of the integral interpolation. In the following, the numerical results shown are obtained through the SPH formulations detailed in [5] and [6].

PROTOTYPE PROBLEM 1: STRETCHING OF A FREE-SURFACE CIRCULAR FLUID PATCH

The first prototype problem considered is the compression on its diameter of an initially circular fluid domain. The full incompressible-flow analytical solution has been derived, both in two and three dimensions, and is used as reference solution for our SPH model. The results of this solver are analyzed in terms of kinematics, which is standard, and in terms of pressure solution, which is more unusual. At the initial instant, the fluid domain Ω is a two-dimensional fluid ball of radius R surrounded by the void and subjected to the velocity and pressure fields (A_0 is a constant)

$$u_0(x,y) = -A_0 x \; ; \quad v_0(x,y) = A_0 y \quad \text{and} \quad p_0(x,y) = \frac{\rho A_0^2}{2}\left[R^2 - (x^2+y^2)\right] \tag{1}$$

which satisfy the Poisson equation for the pressure $\nabla^2 p_0 = -2\rho A_0^2$. With such an initial condition the domain Ω preserves an elliptical form during the whole evolution (see left part of figure 1). A convergence study has first been handled on the kinematic quantities, namely the velocity at given points in the flow, and the conservation of the kinematic energy. To this purpose, four numbers N of particles have been employed, ranging from 1250 to 80000 particles spread in the fluid domain. The convergence obtained is linear for both quantities, and the mean errors measured fall below 1% with 20000 particles.

In the simulations, with the coherent pressure/velocity fields which are initially applied (*i.e.* satisfying the pressure Poisson equation), it is expected that the obtained pressure solution will be identical to the incompressible analytical solution (no velocity divergence initially, and no source of divergence then). Nonetheless, some oscillations around the analytical solution do appear in the SPH result, due to cumulation of numerical errors. Therefore, a distinction must be made between two aspects of the SPH often mixed together. Actually, this method simulates a weakly-compressible

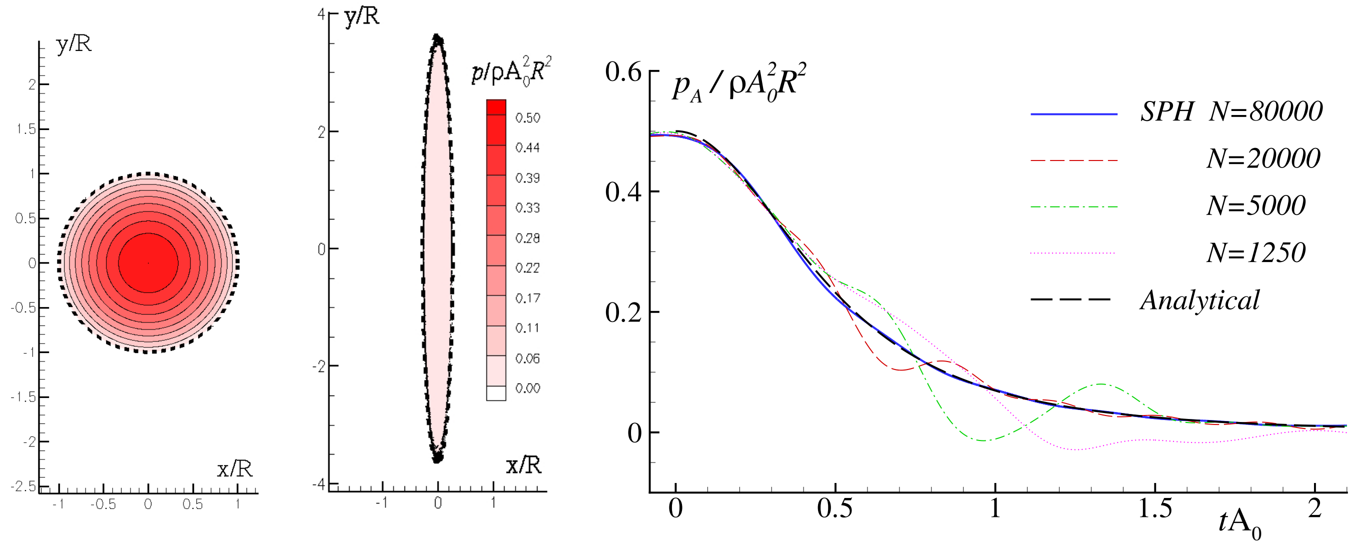

Figure 1: Evolution of an initially circular fluid patch. Left: SPH solution at initial and final instants ($t\,A_0 = 2{\cdot}00$). The color contours are those of the nondimensional pressure field $p/\rho\,A_0^2\,R^2$. The dashed lines represent the analytical incompressible-flow free surfaces. Right: filtered pressure evolution at the center of the fluid domain.

flow, and therefore provides solutions where both the incompressible and compressible components of the flow are present. However, the compressibility in SPH is used only in the weak-compressible domain, that is the range where local Mach numbers are sufficiently low to be able to make the incompressible-flow assumption. And in that sense, the acoustic oscillations found in the solution provided by the SPH model are not physical. Therefore, they can be filtered out, easily because they belong to a range of frequencies that is much higher than the one of the incompressible flow solution. Having filtered out those frequencies (*cf.* right plot in figure 1), the found convergence rate on the local pressure is greater than 1, the final mean error (for 80000 particles) being of 1·6%.

▶ **Three-dimensional ellipsoidal fluid patch** The same kind of prototype test comparison can be handled in three dimensions as well, where again we have derived the analytical solution. The figure 2 shows half of the fluid domain at the initial and final stages. The SPH solution (pressure colored particles) is compared to the analytical free surface (shown by two meridians and the equator). For that run, $\simeq 1\,000\,000$ particles are employed. The CPU cost per time step is $\simeq 30$s on a 3·2-GHz Xeon single-processor PC; with 4700 iterations necessary to reach $tA_0 = 1{\cdot}0$. Errors on the kinematic quantities are very comparable to those found in two dimensions for the same discretization. As well, for the pressure solution, the same conclusions can be drawn as those in two dimensions.

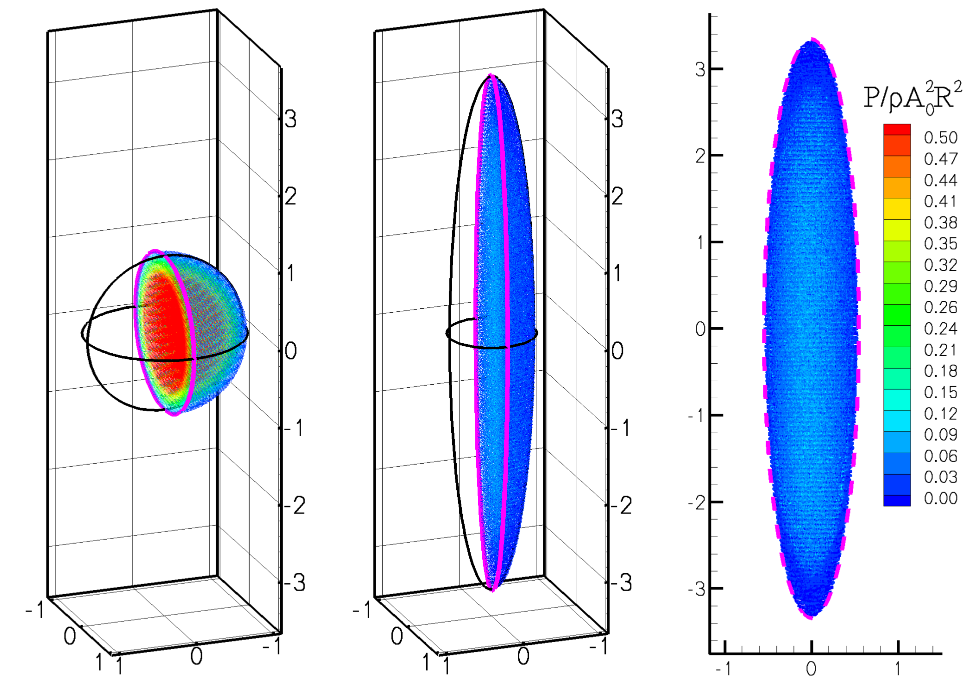

Figure 2: Evolution of an initially spherical fluid patch: initial and final ($tA_0 = 1{\cdot}0$) SPH solutions. The color contours are those of the nondimensional pressure field. The lines represent the meridians and equator of the analytical incompressible-flow free surface. The right plot shows a vertical central cut of the fluid domain at the final stage.

PROTOTYPE PROBLEM 2: ROTATION OF A FREE-SURFACE SQUARE FLUID PATCH

The second prototype problem considered is the constant rotation of an initially squared fluid domain. Comparison is made to a Boundary Element Model by Greco [7]. This time, the initial fluid domain Ω is a two-dimensional square of side L, still surrounded by the void, and subjected initially to the rigid-rotation field

$$u_0(x,y) = \omega y \ ; \quad v_0(x,y) = -\omega x \quad \text{and} \quad \nabla^2 p_0 = 2\rho\omega^2 \tag{2}$$

The initial pressure is obtained as a series expansion by solving the Poisson equation of (2), see figure 3. The center of rotation $(0;0)$ is referred to as point A in the following, and the angular velocity ω is a constant. Regarding the flow evolution, this test is the opposite of the previous one, and will thus make appear other kinds of difficulties. Actually, the stretching of the circular patch was characterized by a positive pressure field inside the domain, leading to a compression / expansion evolution naturally keeping the fluid domain compact. Conversely here, the square patch of fluid is submitted to a centrifugal force, therefore a negative pressure field which is well-known to bring tensile instability [8]. As well, large aspect ratios in the deformation will occur, an unphysical numerical fragmentation of the medium becoming a risk, which is avoided using especially the tensile stability control proposed by Monaghan [9]. The SPH results are globally in nice agreement with the BEM free-surface solution, even at the late stages of the evolution (see figure 3) when the fluid domain is highly deformed, mainly composed of four thin bent arms. In particular, the four-symmetricity is nicely kept all along the simulations. In the BEM solution, a smoothing of the arm tips is visible, which can be safely attributed to the known difficulties that this kind of models encounters to accurately represent sharp angle evolutions.

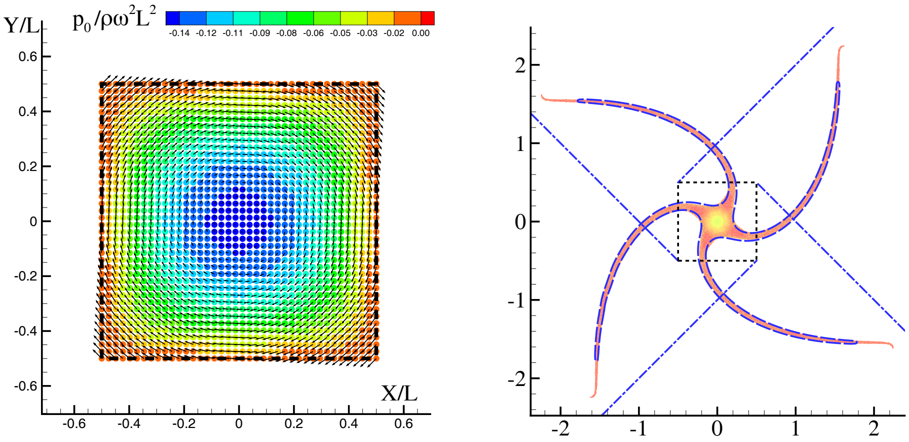

Figure 3: Evolution of an initially square fluid patch. Left: initial velocity (vectors) and pressure (color contours) fields. Right: comparison of the SPH fluid domain (particles colored with pressure contours) and BEM free surface (dashed line) at time $t\omega = 4{\cdot}0$.

Willing again to assess the quality of our solution in terms of dynamic quantities, the pressure solution has been examined at point A. Figure 4 shows the pressure time histories calculated there by both the SPH and BEM methods. The standard SPH solution oscillates near the incompressible BEM solution, but its mean evolution is slightly lower. The characteristic frequency of these oscillations can be related to the vibration mode excited, which is theoretically given by $f_0 = c/(\sqrt{2}L) = 4{\cdot}95\omega$ at $t = 0$ (cf. [10]). To better understand these discrepancies, we compared the standard SPH solution to the one of a second-order Lagrangian Finite Difference Method (FDM), that we have developed (not shown here). This demonstrated that the spurious oscillations present in the standard SPH pressure solution do not originate from the weak-compressibility itself, but from the rather low precision of the SPH integral interpolation. Therefore, a higher-order SPH model has been implemented, and actually its pressure solution shows to be much closer to the BEM solution (*cf.* figure 4). Using such a higher-order interpolation scheme as regular model could have a number of interests in terms of regularity and accuracy of the solution. Conversely, this kind of solution is more CPU-time consuming (matrix inversions, searching for the boundary, remeshing, etc.) and less robust (non conservative, completely Lagrangian).

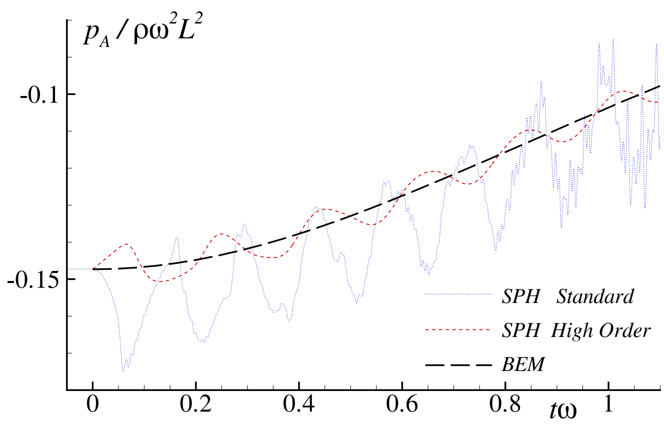

Figure 4: Evolution of an initially square fluid patch: pressure time history at point A. Comparison of the (weakly-compressible flow) standard and higher-order SPH solutions to the incompressible-flow BEM solution.

PROTOTYPE PROBLEM 3: VORTICAL EVOLUTION OF A FREE-SURFACE SQUARE FLUID PATCH

The last test consists in another free-surface square patch of fluid, but this time applying initially a non-uniform vorticity field instead of the constant one that was given in the last example. The interest of such a change in the forcing terms is that it will induce a more complex kinematics of the flow, leading not only to large stretching deformations, but also to distortion of the flow and free-surface reconnection. The initial field is this time

$$\boldsymbol{u}_0(\boldsymbol{x}) = \left[V_0\left(f(y) - e^{-4}\right); -V_0\left(f(x) - e^{-4}\right)\right] \quad \text{with} \quad f(x) = e^{-\left(4x/L\right)^2} \ ; \quad \text{and} \quad \nabla^2 p_0 = 2\rho\left(\frac{32V_0}{L^2}\right)^2 f(x)f(y) \tag{3}$$

where V_0 is a constant. Again, the initial pressure derives from the resolution of the Poisson equation in (3). This time, a BEM solver cannot be used since some features of the flow (the reconnection of the free surface in particular) cannot be handled by such models, at least in their regular form. A more sophisticated solver has therefore been employed, which is a two-phase incompressible-flow Eulerian FDM (EFDM) model combined with a Level-Set algorithm to capture the interface, developed by Colicchio [11]. The SPH model is in good agreement with this Level-Set EFDM solver all along the evolution, as it is visible in figure 5. The resulting flow is very complex and leads to pronounced deformations of the free surface that breaks and reconnects at $t \simeq 1{\cdot}2L/V_0$. It can be noticed that both codes nicely capture this reconnection and then predict very similar symmetric cavities. The little differences found on the details of the solutions, such as the ends of the arms or the size of the cavities, could be attributed to the presence of the second phase in the Level-Set EFDM solver.

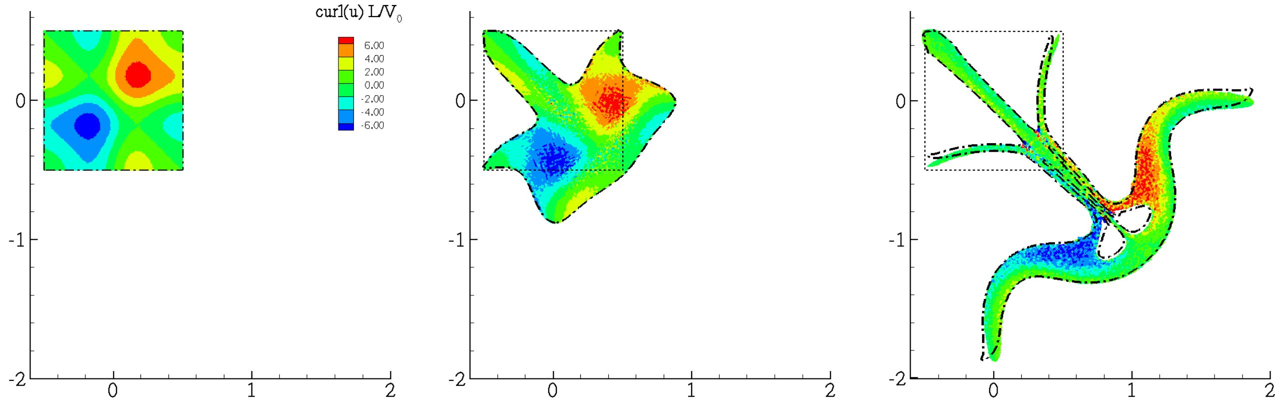

Figure 5: Evolution of an initially square fluid patch with vorticity: comparison to a Level-Set EFDM solver at times $tV_0/L = 0, 0{\cdot}4$ and $1{\cdot}5$. The SPH particles are colored with the vorticity levels. The dashed-dot lines represent the free-surface evolution predicted by the Level-Set EFDM solver.

CONCLUSION

In this paper a family of prototype tests has been proposed, believed to be suitable to study in depth various features of free-surface particle methods (and more generally, of any free-surface solver). Namely, the investigated tests have been chosen to highlight different characteristics of the SPH method little considered in literature, their increasing complexity being meant to discuss of more and more critical aspects. For some of these tests, the full analytical solution has been found; for others comparison has been made to various solvers. The same kind of tests can also be useful to investigate three-dimensional versions of SPH solvers. Moreover, it has been established that the high-frequencies found in the local load signals by the standard SPH derive from numerical errors in the interpolation integral. Those acoustic frequencies are therefore spurious, and do not reflect a physical effect of the weak-compressibility assumed in the method. Finally, it has been proved that, using adequate numerical tools and taking care of the dynamic part of the solution, the SPH method is able to get through these critical tests, with a very satisfactory level of accuracy. In particular, its convergence has been heuristically evidenced.

Acknowledgements The present research activity is developed in the framework of the "Programma Sicurezza" funded by the Italian *Ministero Infrastrutture e Trasporti*. This work has also been supported by the *Délégation Générale pour l'Armement* (DGA) of the French Ministry of Defense under post-doctorate funding n°9860830054.

REFERENCES

[1] M. Landrini, A. Colagrossi & M.P. Tulin, Breaking bow and stern waves: numerical simulations, Proc. 16^{th} Int. Workshop on Water Waves and Floating Bodies, Hiroshima, Japan, 2001.
[2] M. Landrini, A. Colagrossi & M.P. Tulin, A novel SPH formulation for 2-phase flows, Proc. 17^{th} Int. Workshop on Water Waves and Floating Bodies, Cambridge, U.K., 2002.
[3] A. Colagrossi, C. Lugni, M. Greco & O.M. Faltinsen, Experimental and numerical investigation of 2D sloshing with slamming, Proc. 19^{th} Int. Workshop on Water Waves and Floating Bodies, Cortona, Italy, 2004.
[4] M. Doring, G. Oger, B. Alessandrini & P. Ferrant, SPH simulations of floating bodies in waves, Proc. 19^{th} Int. Workshop on Water Waves and Floating Bodies, Cortona, Italy, 2004.
[5] A. Colagrossi & M. Landrini, Numerical simulation of interfacial flows by Smoothed Particle Hydrodynamics, J. Comput. Phys. 191 (2003) 399–406.
[6] A. Colagrossi, A meshless Lagrangian method for free-surface and interface flows with fragmentation, PhD Thesis, University of Rome La Sapienza (Italy), 2005.
[7] M. Greco, A two-dimensional study of green-water loading, PhD Thesis, University of Trondheim (Norway), 2001.
[8] J.W. Swegle, D.L. Hicks & S.W. Attaway, Smoothed Particle Hydrodynamics stability analysis, J. Comput. Phys. 116 (1995) 123–134.
[9] J.J. Monaghan, SPH without a tensile instability, J. Comput. Phys. 159 (2000) 290–311.
[10] P.M. Morse & K. Uno Ingard, Theoretical acoustics, Princeton University Press (1986) 211.
[11] G. Colicchio, Violent disturbance and fragmentation of free surfaces, PhD Thesis, University of Southampton (UK), 2004.